CHAPTER 4

POWER SAWS AND DRILLING MACHINES

CHAPTER LEARNING OBJECTIVES

Upon completing this chapter, you should be able to do the following:

- *Identify and explain the use of power saws.*
- *Identify and explain the use of drilling machines.*
- *Explain the use of twist drills.*

A machine tool is a power driven machine that holds the material and cutting tool and brings them together so the material is drilled, cut, shaved or ground. Chapter 1 covered common handtools. In this chapter we will deal primarily with power saws and drilling machines.

POWER SAWS

Before we discuss the operation of power saws, you must realize the importance of observing safety precautions. Carelessness is one of the prime causes of accidents in the machine shop. Moving machinery is always a potential danger. When this machinery is associated with sharp cutting tools, the hazard is greatly increased. As with any shop equipment you must observe all posted safety precautions. Review your equipment operators manual for safety precautions and any chapters of *Navy Occupational Safety and Health (NAVOSH) Program Manual for Forces Afloat,* OPNAV Instruction 5100.19B, that pertain to the equipment.

POWER HACKSAW

The power hacksaw is found in many Navy machine shops. It is used to cut bar stock, pipe, tubing, or other metal stock. The power hacksaw consists of a base, a saw frame, and a work holding device. Figure 4-1 shows a standard power hacksaw.

The base consists of a reservoir to hold the coolant, a coolant pump, the drive motor and a transmission for speed selection. Some models may have the feed mechanism attached to the base.

The saw frame consists of linkage and a circular disk with an eccentric (off center) pin designed to convert circular motion into reciprocating motion. The blade is inserted between the two blade holders and securely attached by either hardened pins or socket head screws. The inside blade holder is adjustable. This adjustable blade holder allows the correct tension to be put on the blade to ensure that it is held rigidly enough to prevent it from wandering and causing a slanted cut. The feed control

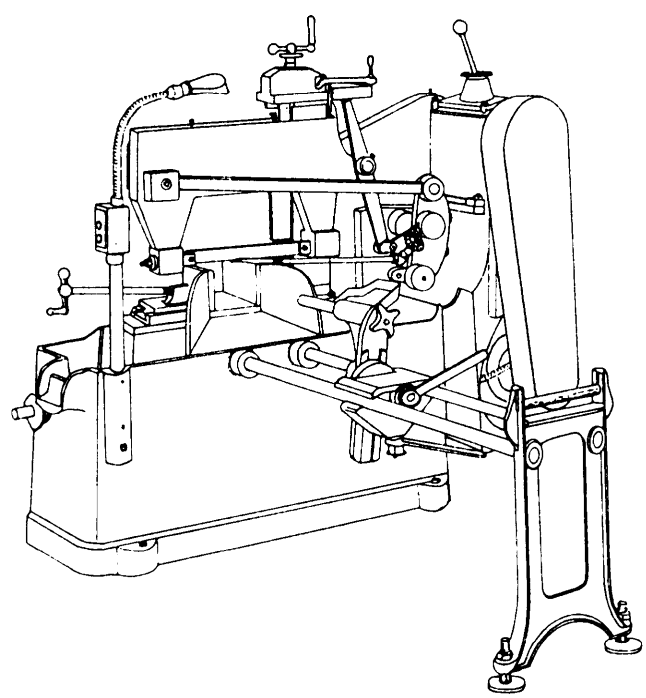


Figure 4-1.—Standard power hacksaw.

mechanism is also attached to the saw frame on many models.

The work holding device is normally a vise with one stationary jaw and one movable jaw. The movable jaw is mounted over a toothed rack to permit a rapid and easy initial adjustment close to the material to be cut. Do final tightening by turning the vise screw until the material is held securely. An adjustable stop permits pieces of the same length to be cut without measuring each piece separately. A stock support stand (available for both sides of the saw) keeps long stock from falling when being cut.

The power hacksaw illustrated can handle material up to 4 inches wide and 4 inches thick.

Blade Selection

The blade shown in figure 4-2 is especially designed for use with the power hacksaw. It is made with a tough alloy steel back and high-speed steel teeth. This combination gives a strong blade and cutting edge suitable for high-speed sawing.

These blades differ by the pitch of the teeth (number of teeth per inch). You should select the correct pitch of teeth for a particular job according to the size and material composition of the section to be cut. Use coarse pitch teeth for wide, heavy sections to provide ample chip clearance. For thinner sections, use a blade with a pitch that keeps two or more teeth in contact with the work so that the teeth do not straddle the work. Straddling strips the teeth from the blade. In general, select blades according to the following information:

- Coarse (4 teeth per inch), for soft steel, cast iron, and bronze.
- Regular (6 to 8 teeth per inch), for annealed high carbon steel and high-speed steel.
- Medium (10 teeth per inch), for solid brass stock, iron pipe, and heavy tubing.
- Fine (14 teeth per inch), for thin tubing and sheet metals.

Coolant

You should use a coolant for most power hacksawing operations. (Cast iron can be sawed dry.) The coolant keeps the kerf (narrow slot created by the cutting action of the blade) clear of chips so that the blade does not bind up and start cutting crooked. The coolant protects the teeth from overheating. This allows you to increase the rate of cutting beyond the speed possible without coolant. A soluble oil solution with a mixture of the oil and water will be suitable for most sawing operations. The normal mixture for soluble oil is 40 parts water to 1 part oil. You also may use a synthetic coolant.

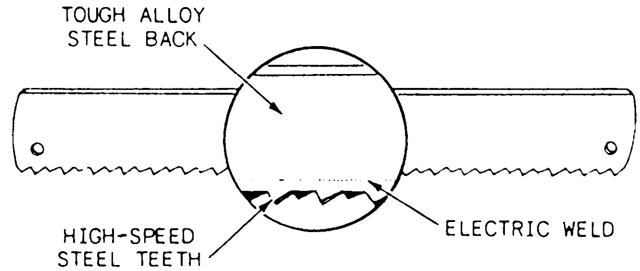


Figure 4-2.—Hacksaw blade.

Feeds and Speeds

A power hacksaw will have one of three types of feed mechanisms:

- Mechanical feed, which ranges from 0.001 to 0.025 inch per stroke, depending upon the class and type of material being cut
- Hydraulic feed, which exerts a constant pressure but is designed so that the feed is automatically stopped or shortened at hard spots to decrease the pressure on the saw until the hard spot has been cut through
- Gravity feed, in which weights are placed on the saw frame and shifted to give more or less pressure of the saw blade against the material being cut

To prevent unnecessary wear on the back sides of the saw blade teeth, the saw frame and blade are automatically raised clear of the surface being cut on each return stroke. The rate of feed or the pressure exerted by the blade on the cutting stroke depends on several factors—the toughness and hardness of the material, the size, and in the case of a hollow pipe, the wall thickness. You must cut a hard, large diameter piece of stock with a slower or lighter feed rate than you would use with a soft, small diameter piece of stock. Cut pipe with thin walls with a relatively light feed rate. This prevents stripping the teeth from the saw blade or collapsing the walls of the pipe. A feed rate that is too heavy or fast will often cause the saw blade to wander, producing an angled cut.

The speed of hacksaws is stated in strokes per minute, counting only those strokes on which the blade comes in contact with the stock. You will use a gear lever to change speeds. There may be a chart attached to or near the saw, giving recommended speeds for cutting various metals. You may use the following speeds:

- Medium and low carbon steel, brass, and soft metals—136.
- Alloy steel, annealed tool steel, and cast iron—90.
- Unannealed tool steel, and stainless steel—60.

Saw Operation

A power hacksaw is relatively simple to operate. There are, however, a few checks you should make to ensure good cuts. Support overhanging ends of long pieces to prevent sudden breaks at the cut before the work is completely cut through. Block up irregular shapes so that the vise holds firmly. Check the blade to be sure it is sharp and that it is secured at the proper tension.

Place the workpiece in the clamping device, adjusting it so the cutting off mark is in line with the blade. Turn the vise lever to clamp the material in place. Be sure the material is held firmly.

See that the blade is not touching the workpiece when you start the machine. You often break blades when you don't follow this rule. Feed the blade slowly into the work, and adjust the coolant nozzle so that it directs the fluid over the saw blade.

CONTINUOUS FEED CUTOFF SAW

Figure 4-3 illustrates a type of cutoff saw that is now being used in most shops throughout the Navy. There are different models of this saw, but the basic design and operating principles remain the same.

Deleted—No permission granted for electronic copy.

Figure 4-3.—Continuous feed cutoff saw.

Deleted—No permission granted for electronic copy.

Figure 4-4.—Control panel (DoAll saw).

Band Selection and Installation

The bands for the continuous feed cutoff saw are nothing more than an endless hacksaw blade. With this thought in mind, you can see that all the factors that were discussed for power hacksaw blade selection can be applied to this saw. This saw is also equipped with a job selector (fig. 4-3) to help you make the proper selection. The bands come in two different forms; ready made loops of the proper length and coils of continuous lengths of 100 feet or more. You may not change the presized band, but you must cut the coils of saw bands to the proper length and then butt-welded the ends. (Butt welding is covered later in this chapter.) Once you have selected the saw band, install it following the instructions in your saws operators manual.

Saw Operation

You will control the movement of the saw head from the control panel (fig. 4-4). Once you are ready to operate the saw follow these steps:

1. Pull the hydraulic start button (this will pressurize both the hydraulic and coolant systems).
2. Adjust the vice for work size and workstop for the desired cut-off length. Place it’s band speed, feed and coolant at approximate settings.
3. Raise the head with the head control joy stick, then set the control on “hold”.
4. Position the workpiece and clamp the vice.
5. Pull the “Band Start” button and turn the head control to the “saw” position to start the cut. Adjust coolant, band speed, and feed if necessary.
6. The machine stops automatically at the end of the cut.

METAL CUTTING BANDSAWS

Metal cutting bandsaws are standard equipment in repair ships and shore repair facilities. You can use these machines for nonprecision cutting similar to power hacksaws. You can use some types for precision cutting, filing, and polishing. A bandsaw is more flexible for straight cutting than a power hacksaw in that it can cut objects of any reasonable size and of regular and irregular shapes.

Figure 4-5 illustrates a metal cutting bandsaw with a tiltable table. On the type shown, you should feed the work either manually or by power to the blade which runs in a fixed position.

The tiltable band saw is particularly suited to straight and angle cuts on large, long, or heavy pieces.

The tiltable table is convenient for contour cutting because you can quickly change the angle at which you feed work to the blade. This machine usually has special attachments and accessories to make precision inside or outside cuts of contours and disks and to miter. It has special bands to file and polish work. The saw bands, file bands, and polishing bands used on these machines are called BAND TOOLS, and the machine itself is often referred to as a BAND TOOL MACHINE. We have listed some definitions that will

Deleted—No permission granted for electronic copy.

Figure 4-5.—Tiltable (contour) metal-cutting bandsaw.

help you understand band tool terminology for saws, files, and polishing bands.

Saw Bands

A saw band has the following characteristics which are illustrated in figures 4-6 through 4-8.

PITCH: The number of teeth per linear inch.

WIDTH: The distance across the flat face of the band. The width measurement is always expressed in inches, or fractions of an inch.

GAUGE: The thickness of the band back. This measurement is expressed in thousandths of an inch.

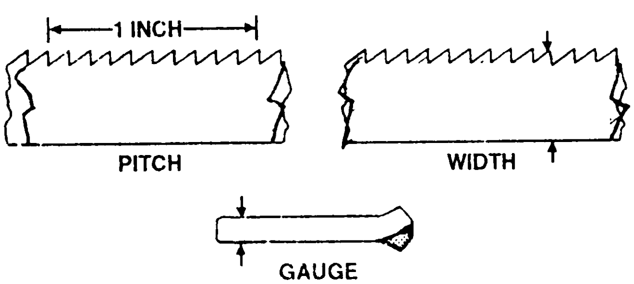


Figure 4-6.—Pitch, width, and gauge.

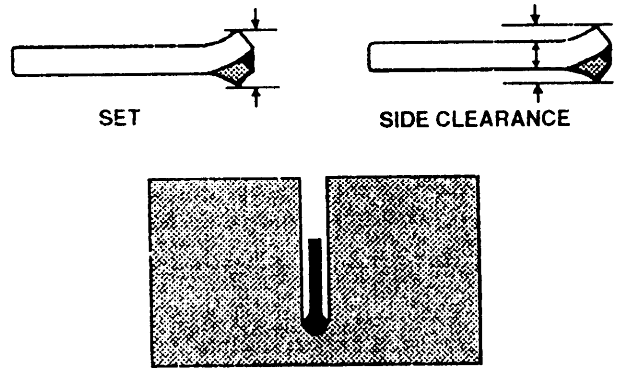


Figure 4-7.—Set and side clearance.

SET: The bend or spread given to the teeth to provide clearance for the body or band back when you make a cut.

SIDE CLEARANCE: The difference between the dimension of the band back (gauge) and the set of the teeth. Side clearance provides running room for the band back in the kerf or cut. Without side clearance, a band will bind in the kerf.

SET PATTERN: One of three distinct patterns (raker, wave, and straight) in which teeth are set. Raker set bands are generally used for solid cross section work. Wave set bands are used to cut hollow materials, such as pipe and tubing, and for other work where there is a great deal of variation in thickness. Straight set bands are not used to any great extent to cut metal.

TEMPER: The degree of hardness of the teeth, indicated by the letters A and B, temper A being the harder. Temper A bands are used for practically all bandsaw metal cutting work.

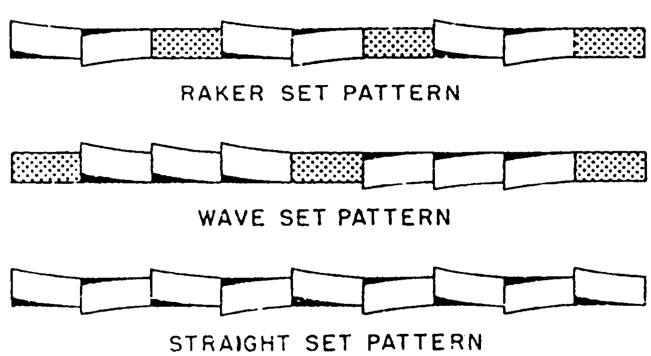


Figure 4-8.—Set pattern.

Deleted—No permission granted for electronic copy.

Figure 4-9.—File band flexing principle and construction.

File Bands

A file band consists of a long steel strip that has a number of file segments mounted on it. It can be flexed around the band wheels and still present a straight line at the point of work. Figure 4-9 shows the file band flexing principle and the construction of a file band.

Polishing Bands

Abrasive coated fabric bands are used for grinding and polishing operations in a band tool machine. They are mounted in the same way as saw and file bands. Figure 4-10 shows a polishing band. Figure 4-11 shows a backup support strip being installed, before the polishing band is installed.

Band Tool Guides

As an MR you will primarily use three types of band tool guides. They are saw band guides, file band guides and polishing band guides.

SAW BAND GUIDES.—The upper and lower guides keep the saw band in its normal track when when you apply work pressure to the saw. The lower guide is in a fixed position under the work table, and the upper guide is attached to a vertically adjustable arm above the table which permits you to raise or lower the guide to suit the height of work. To obtain adequate support for the band and yet not interfere

Deleted—No permission granted for electronic copy.

Figure 4-10.—Polishing band.

Deleted—No permission granted for electronic copy.

Figure 4-11.—Installing a backup support strip for polishing band.

with the sawing operation, place the upper guide so that it will clear the top of the workpiece by 1/8 to 3/8 of an inch. Figure 4-12 shows the two principal types of saw band guides: the insert type and the roller

Deleted—No permission granted for electronic copy.

Figure 4-13.—File band guide.

type. Note in both types the antifriction bearing surface for the band's relatively thin back edge. This feature allows the necessary work pressure to be placed on the saw without causing serious rubbing and wear. Be sure to lubricate the bearings of the guide rollers according to the manufacturer's recommendations.

FILE BAND AND POLISHING BAND GUIDES.—For band filing operations, replace the regular saw band guide with a flat, smooth-surface metal backup support strip, as shown in figure 4-13.

Deleted—No permission granted for electronic copy.

Figure 4-12.—Saw band guides.

This prevents sagging of the file band at the point of work. Use a similar support for a polishing band. This support has a graphite-impregnated fabric face that prevents undue wear on the back of the polishing band, which also is fabric.

Selection Of Saw Bands, Speeds And Feeds

Saw bands are available in widths ranging from 1/16 to 1 inch; in various even-numbered pitches from 6 to 32; and in three gauges—0.025, 0.032, and 0.035 inch. The gauge of saw band that can be used in any particular machine depends on the size of the band wheels. You cannot sucessfully use a thick saw band on a machine that has small diameter bandwheels; therefore, only one or two gauges of blades may be available for some machines. Generally, only temper A, raker set, and wave set bands are used to cut metal.

Saw bands come in different forms for different working. Some are furnished in ready made loops of the correct length. Others come in coils of 100 feet or more from which you must cut a length and form it into a band loop by butt welding the ends together in a special machine. Later in this chapter, we will explain the process of joining the ends and installing bands.

Band tool machines have a multitude of band speeds, ranging from about 50 feet per minute to about 1500 feet per minute. Most of them are equipped with a hydraulic feed which provides three feeding pressures: low, medium, and heavy.

Success in precision sawing with's metal cutting bandsaw depends on several factors: You must select the correct saw blade or band, you must run the saw band at the correct speed, and you must feed the work to the saw at the correct rate. Many band tool machines have a job selector similar to the one shown in figure 4-14. It indicates the kind of saw band you should use, the speed at which you should operate the machine, and the power feed pressure you should use to cut various materials.

TOOTH PITCH.—Tooth pitch is the primary consideration in selecting a saw band for any cutting job. To cut thin materials, the pitch should be fine enough so that at least two teeth are in contact with the work; fewer than two may cause the teeth to snag and tear loose from the band. To cut thick material, you should not have too many teeth in contact with the work, because as you increase the number of teeth in contact, you must increase the feed pressure in order to force the teeth into the material.

Excessive feed pressure puts severe strain on the band and the band guides. It also causes the band to wander sideways which results in off-line cutting.

Other points to consider in selecting a saw band of proper pitch are the composition of the material to be cut, its hardness, and its toughness.

BAND WIDTH AND GAUGE.—The general rule is to use the widest and thickest saw band that can do the job successfully. For example, you should use a band of maximum width and thickness (if bands of different thickness are available) when the job calls for only straight cuts. On the other hand, when a layout requires radius cuts (curved cuts), select a band that can follow the sharpest curve involved. For curved work, select the widest band that will negotiate the smallest radius required. The saw band width selection guides, shown in figure 4-15, give the radius of the sharpest curve that can be cut with a particular width saw band. Note that the job selector illustrated

Deleted—No permission granted for electronic copy.

Figure 4-14.—Job selector.

Deleted—No permission granted for electronic copy.

Figure 4-15.—Saw band width selection guides.

in figure 4-14 contains a saw band radii cutting diagram similar to the one shown in figure 4-15.

BAND SPEEDS.—The rate at which the saw band travels in feet per minute from wheel to wheel is the saw band velocity. Saw band velocity has considerable effect upon both the smoothness of the cut surfaces and the life of the band. The higher the band velocity, the smoother the cut; however, heat generated at the cutting point increases as band velocity increases. We cannot cover the details of adjustment because different machines use different methods. Consult the manufacturer's technical manual for your particular machine and learn how to set up the various speeds available.

FEEDS.—Though manual feeding of the work to the saw is satisfactory to cut metals up to 1 inch thick, power feeding generally provides better results and will be much safer for the operator. Regardless of whether you use power or manual feed, it is important not to crowd the saw because the band will tend to bend and twist. However, feed pressure must not be so light that the teeth slip across the material instead of cutting through because this rapidly dulls the teeth. The job selector, shown in figure 4-14, shows the correct feed pressures to cut any of the materials listed on the outer ring of the dial. In the absence of a job selector, you can use table 4-1 as a guide to select feed pressures for hard, medium hard, and soft metals.

The power feed controls vary with different makes of bandsaws and even with different models of the same make; therefore, we will not describe them here. Consult the manufacturer's technical manual and study the particular machine to learn its power feed arrangement and control.

Sizing, Splicing, And Installing Bands

Most contour cutting type bandsaws are provided with a buttwelder-grinder combination, you should use it to join saw bands that come in bulk stock coil form, and to join broken band loops. The butt welder is usually attached to the saw machine, as shown in

Table 4-1.—Feed Pressures for Hard, Medium Hard, and Soft Metal

Material	Work thickness				
	0-1/4"	1/4-1/2"	1/2-1"	1-3"	Over 3"
Tool Steel	M	M	H	H	H
Cast iron	M	M	M	H	H
Mild steel	L	M	H	H	H
Nickel-copper	L	M	H	H	H
Copper-nickel	L	L	M	H	H
Zinc	L	L	M	M	M
Lead	L	L	M	M	M

*

L-light, M-medium, H-heavy.

Deleted—No permission granted for electronic copy.

Figure 4-16.—Butt welder-grinder unit.

figure 4-16, but it may be portable or a pedestal-mounted accessory. The butt welder also makes inside cutting possible, since the saw band loop can be parted and rejoined after having been threaded through a starting hole in the work.

The following sections describe how to determine the length of the band, how to join the ends in the butt welder, and how to install a band tool in the machine.

BAND LENGTH.—To determine the correct saw band length for any two-wheeled bandsaw, measure the distance from the center of one wheel to the center of the other wheel, multiply by 2, and add the circumference of one wheel.

The easiest way to determine the length of saw band for a three-wheeled machine is to take a steel tape, thread it through the machine over the wheels, and measure the distance.

Before measuring between the wheel centers, adjust the upper wheel so that it is approximately halfway between the upper and lower limits of its vertical travel. This allows you to take up any band stretch.

BAND SPLICING.—Figure 4-16 shows band ends being joined by using a butt welder. Use the following procedure:

1. Grind both ends of the band until they are square with the band back edge. If you do not do this carefully, the weld may not go completely across the ends of the band and, as a result, the weld will not withstand the pressure of the cutting. One easy method to ensure that the ends of the band will go together perfectly is to twist one end 180 degrees and then place the band ends on top of each other. This will provide a set of teeth and a band back edge on both sides of the stacked ends. Ensure that the band back edge and the teeth are in a straight line on both sides. Carefully touch the tips of the ends of the band to the face of the grinding wheel and lightly grind until both ends have been ground completely across. Release the ends of the band so that they assume their normal position. Lay the back edge of the band on a flat surface and bring the ends together. If you did the grinding correctly, the ends will meet perfectly.

2. Set the controls of the butt welder to the weld position and adjust the adjusting lever according to the width of band to be welded. The various models of butt welders differ in the number of controls that must be set and the method of setting them. Most models have a lever that must be placed in the weld position so that the stationary and the movable clamping jaws are separated the correct distance. Some models have a resistance setting control that is set according to the width of the band, while other models have a jaw pressure control knob that is also set according to band width. Read the manufacturer's instruction manual before welding.

3. Place the ends of the band in the jaws with the teeth of the band facing away from the welder. Push the back edge of the band firmly back toward the flat surfaces behind the clamping jaws to ensure proper alignment. Position the ends of the band so that they touch each other and are located in the center of the jaw opening. Some models of butt welders have interchangeable inserts for the clamping jaws to permit welding bands of different widths. This is done so that the teeth of the band are not damaged when the jaws are clamped tight.

4. You are now ready to weld the band. Some welders require that the weld button be fully depressed and held until the welding is complete, while other welders required only that the button be fully depressed and then quickly released. There will be a shower of sparks from the welding action. Be

sure you are wearing either safety glasses or a face shield before welding and then stand back from the welder when you push the button.

5. When the welding is complete, release the jaw clamps and remove the band from the welder. Inspect the band to be sure it is straight and welded completely across. Do not bend or flex the band at this time to test the weld. The welding process has made the weld and the area near it hard and brittle and breakage will probably occur.

6. Place the lever that controls movement of the jaws in the anneal position. This should separate the jaws again. Set the control that regulates the anneal temperature to the setting for the width of the band.

7. Place the band in the clamping jaws with the teeth toward the welder and the welded section in the center of the jaw opening. Close the jaws.

8. The band is ready to be annealed. Push and then quickly release the anneal button repeatedly until the welded area becomes a dull cherry red. (Do NOT push and hold the anneal button. This will overheat and damage the band.) After the proper temperature is reached, push the anneal button and release it with increasingly longer intervals between the push cycle to allow the band to cool slowly.

9. The metal buildup resulting from the weld must be ground off. Using the attached grinding wheel, remove the weld buildup from both sides and the back of the band until the band fits snugly into the correct slot on the saw band thickness gauge mounted on the welder. Do this grinding carefully to prevent looseness or binding between the saw guides and the band. Be careful not to grind on the teeth of the band.

10. Repeat the procedure for annealing in step 8 after grinding the blade.

11. The weld is complete. To test your weld, hold the band with both hands and form a radius in the band slightly smaller than the smallest wheel on the bandsaw by bringing your hands together. Move your hands up and down in opposite directions and observe the welded area as it rolls around the radius that you formed.

BAND INSTALLATION.—Insert saw band or tool guides of the correct size for the band you are going to install. Adjust the upper band wheel for a height that will allow you to easily loop the band around the wheels. Then place one end of the loop over the upper band wheel and the other end of the loop around the lower band wheel. Be sure that the teeth are pointing downward on the cutting side of the band loop and that the band is properly located in the guides. Place a slight tension on the band by turning the upper wheel takeup hand wheel. Revolve the upper band wheel by hand until the band has found its tracking position. If the band does not track on the center of the crowns of the wheels, use the upper wheel tilt control to correct the band track as illustrated in figure 4-17.

Deleted—No permission granted for electronic copy.

Figure 4-17.—Upper wheel tilt adjustment.

When the band is tracking properly, adjust the band guide rollers or inserts so that you have a total clearance of 0.001 to 0.002 inch between the sides of the band back and the guide rollers or inserts, and a slight contact between the back edge of the band back and the backup bearings of the guides. When you have set the band guide clearance, increase the band tension. The amount of tension to put on the band depends on the width and gauge of the band. A narrow, thin band will not stand as much tension as a wider or thicker band. Too much tension will cause the saw to break; insufficient tension will cause the saw to run off the cutting line. The best way to obtain the proper tension is to start with a moderate tension; if the saw tends to run off the cutting line, increase the tension slightly.

Sawing Operations

We mentioned earlier in the chapter that you can use a band tool machine to do straight, angular, contour, inside, and disk cutting. The following

Deleted—No permission granted for electronic copy.

Figure 4-18.—Work jaw and feed chain adjustment.

paragraphs contain the procedures for each of these cutting operations. But first, let us consider the general rules applicable to all sawing operations.

- Check the level of the work table and adjust the table, if necessary, to suit the angle of the cut.
- Use the proper blade and speed for each cutting operation. This ensures not only the fastest and most accurate work but also longer saw life.
- Always be sure the band guide inserts are the correct size for the width of the band and that they are properly adjusted.
- Before starting the machine, adjust the height of the upper band guide so that it will clear the work from 1/8 to 3/8 inch. The closer the guide is to the work, the greater the accuracy.
- When starting a cut, feed the work to the saw gradually. After the saw has started the kerf, increase the feed slowly to the recommended pressure. Do not make a sudden change in feed pressure because it may cause the band to break.
- Be sure the saw band and guides are properly lubricated.
- Use lubricants and cutting coolants as recommended by the manufacturer of your machine.

STRAIGHT CUTTING WITH POWER FEED.—Most band saws are equipped with hydraulically actuated power feeds. When used, accidental overfeeding is eliminated which greatly extends blade life. To use the power feed:

1. Change band guides as necessary. Select and install the proper band for the job and adjust the band guides.
2. Place the workpiece on the table of the machine and center the work in the work jaw.
3. Loop the feed chain around the work jaw, the chain roller guides, and the left-right guide sprocket, as shown in figure 4-18.

Deleted—No permission granted for electronic copy.

Figure 4-19.—Angular cutting.

Deleted—No permission granted for electronic copy.

Figure 4-20.—Sharp radii cutting eliminated by drilling corner holes.

4. Determine the proper band speed and set the machine speed accordingly.
5. Start the machine and feed the work to the saw in the manner described in the general rules of operation given in the preceding section. Use the left-right control for guiding the work along the layout line.

ANGULAR CUTTING.—Make angular or bevel cuts on flat pieces the same way as straight cuts except you should tilt the table to the desired angle of the cut as shown in figure 4-19.

CONTOUR CUTTING.—You can do contour cutting, that is, following straight, angle, and curved layout lines, offhand or with the power feed. Use the left-right chain control, shown in figure 4-18, to guide the work along the layout line when you use power feed. A fingertip control to actuate the sprocket is located at the edge of the work table. If there are square corners in the layout, drill a hole adjacent to each corner; this will permit the use of a wider band, greater feed pressure, and faster cutting. Figure 4-20 shows the placement of corner holes on a contour cutting layout.

INSIDE CUTTING.—To make an inside cut, drill a starting hole slightly larger in diameter than the width of the band you are going to use. Remove the band from the machine. Shear the band, slip one end through the hole, and then splice the band. When you have done that, reinstall the band and the machine is ready to make the inside cut as illustrated in figure 4-21.

DISK CUTTING.—You can do disk cutting offhand if you lay out the circle on the workpiece and follow the layout circle, or you can use a disk cutting attachment that automatically guides the work so that

Deleted—No permission granted for electronic copy.

Figure 4-21.—Inside cutting.

Deleted—No permission granted for electronic copy.

Figure 4-22.—Disk-cutting attachment.

you cut a perfect circle. Figure 4-22 shows a disk cutting attachment in use. The device consists of a radius arm, a movable pivot point, and a suitable clamp to attach the assembly to the saw guidepost. To cut a disk using this device, lay out the circle and punch a center point. Clamp the radius arm to the guidepost. Position the workpiece (fig. 4-22) so that the saw teeth are tangent to the scribed circle. Adjust the pivot point radially and vertically so that it seats in the center-punch mark; then clamp the pivot point securely. Then rotate the work around the pivot point to cut the disk.

FILING AND POLISHING.—In filing and polish finishing, you will manually feed and guide the work to the band. Proper installation of the guides and backup support strips is very important if you expect good results. A guide fence similar to the one shown in figure 4-23 is very helpful when you work to close tolerances. Be sure to wear goggles or an eye protection shield to file and polish, and above all, be careful of your fingers. For proper band speeds and work pressures, consult the manufacturer's technical manual for the machine you are using.

Deleted—No permission granted for electronic copy.

Figure 4-23.—Polish finishing.

DRILLING MACHINES AND DRILLS

Although drilling machines or drill presses are commonly used by untrained personnel, you cannot assume that you will operat these machines proficiently if you simply insert the proper size drill and start the machine. As a Machinery Repairman, you will be required to perform drilling operations with a great degree of accuracy. Therefore you must be well acquainted with the types of machines and the methods and techniques of operation of drill presses and drills found in Navy machine shops.

A diverse group of people with different training and experience backgrounds use the drill press. Because of that some unsafe operating practices have become routine in spite of the possibility of serious injury. The basic safety precautions for the use of a drill press are listed below:

- Always wear goggles or safety glasses, and wear ear protection when warning signs are posted.
- Keep loose clothing clear of rotating parts.
- **NEVER** attempt to hold a piece being drilled in your hand. Use a vise, hold-down bolts or other suitable clamping device.
- Check the twist drill to be sure that it is properly ground and is not damaged or bent.
- Make sure the cutting tool is held tightly in the drill press spindle.
- Use the correct feeds and speeds.
- When feeding by hand, take care that the drill does not dig in and take an uncontrolled depth of cut.
- Do not remove chips by hand. Use a brush.

The two types of drilling machines or drill presses common to the Navy machine shop are the upright drill press and the radial drill press. These machines have similar operating characteristics but differ in that you position the drilling lead on the radial drill, and you position the workpiece on the upright drill.

UPRIGHT DRILL PRESSES

Upright drill presses discussed in this section will be the general purpose, the heavy duty, and the sensitive drill presses. Nearly all ships have one or more of these types. They are classified primarily by the size of drill that can be used, and by the size of the work that can be set up.

General Purpose Drill Press (Round Column)

Shown in figure 4-24, the general purpose drill press is the most common type of machine found in Navy machine shops. The basic components of this machine are shown in the illustration.

The base has a machined surface with T-slots for heavy or bulky work.

The column supports the work table, the drive mechanism and the spindle head.

The work table and arm can be swiveled around the column and can be moved up or down to adjust for height. In addition, the work table may be rotated 360° about its own center.

The spindle head guides and supports the spindle and can be adjusted vertically to provide maximum support near the spindle socket.

The spindle is a splined shaft with a Morse taper socket to hold the drill. The spline permits vertical movement of the spindle while it is rotating.

The drive mechanism includes the motor, speed and feed change gears, and mechanical controls.

Heavy Duty Drill Press (Box Columns)

The heavy duty drill press (fig. 4-25) is normally used to drill large holes. It differs from the general purpose drill press in that the work table moves only

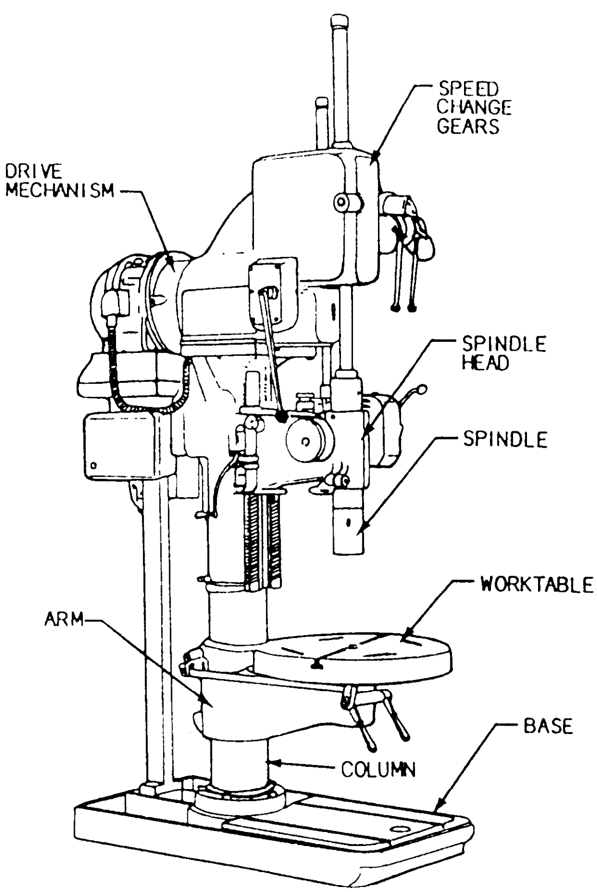
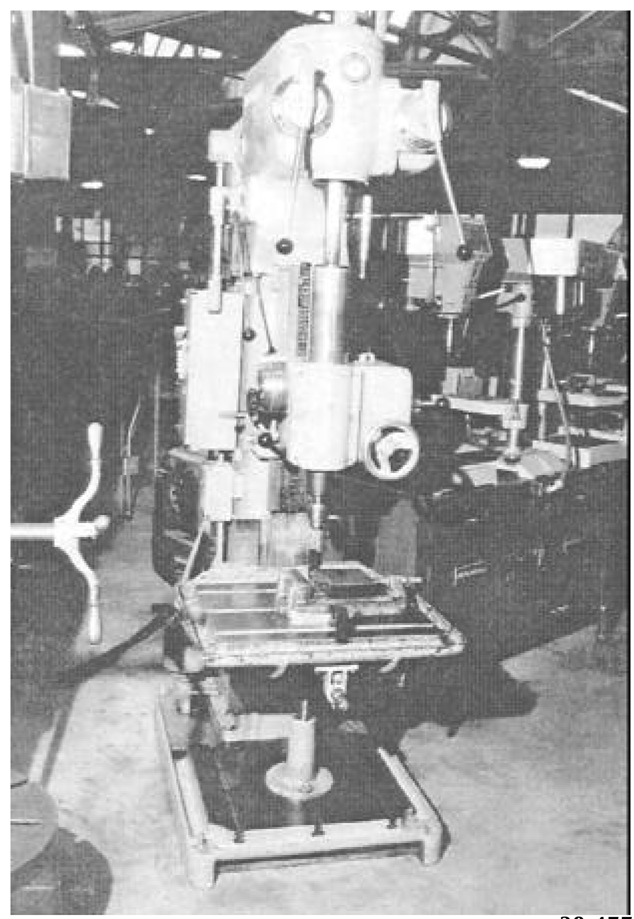


Figure 4-24.—General purpose drill press.



28.477

Figure 4-25.—Heavy duty drill press.

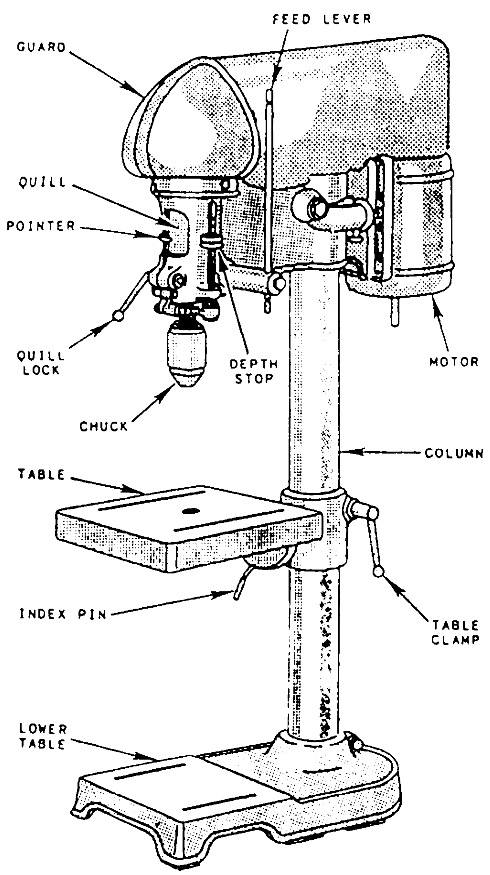


Figure 4-26.—Sensitive drill press.

vertically. The work table is firmly gibbed to vertical ways or tracks on the front of the column and is further supported by a heavy adjusting screw from the base to the bottom of the table. As the table can be moved only vertically, it is necessary to position the work for each hole.

Sensitive Drill Press

The sensitive drill press (fig. 4-26) is used to drill small holes in work under conditions that make it necessary for the operator to "feel" what the cutting tool is doing. The tool is fed into the work by a very simple device-a lever, a pinion and shaft, and a rack that engages the pinion. These drills are nearly always belt-driven because the vibration caused by gearing would be undesirable. Sensitive drill presses are used to drill holes less than one-half inch in diameter. The high-speed range of these machines and the holding devices used make them unsuitable for heavy work.

RADIAL DRILL PRESS

The radial drill press shown in figure 4-27, has a spindle head on an arm that can be rotated axially on

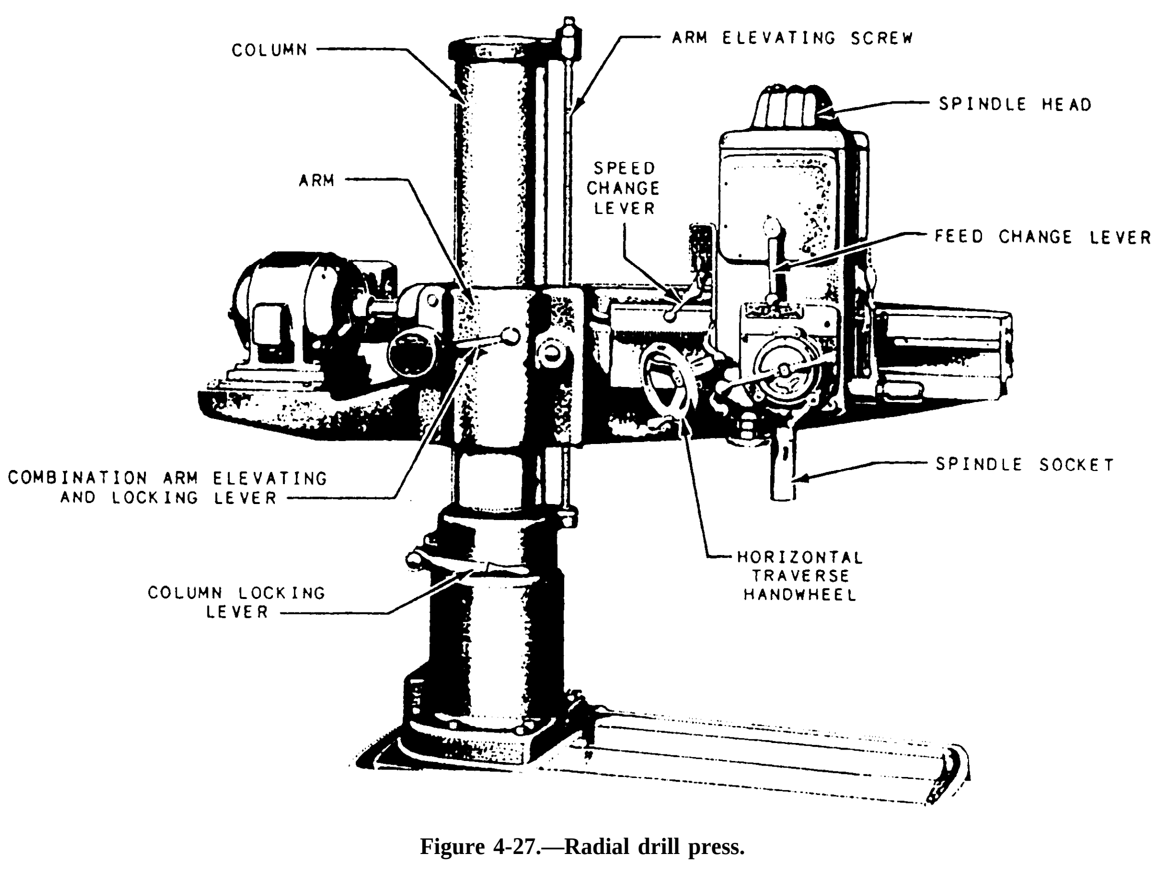


Figure 4-27.—Radial drill press.

the column. The spindle head may be traversed horizontally along the ways of the arm, and the arm may be moved vertically on the column. This machine is especially useful when the workpiece is bulky or heavy or when you need to drill many holes with one setup. The arm and spindle are designed so that the drill can be positioned easily over the layout of the workpiece.

Some operational features that are common to most radial drill presses are: (1) high- and low-speed ranges provided from either a two-speed drive motor or a low-speed drive gear; (2) a reversing mechanism to change the direction of rotation of the spindle by either a reversible motor or a reversing gear in the drive gear train; (3) automatic feed mechanisms that are driven from the spindle and feed the cutting tool at a selected rate per revolution of the spindle; (4) depth setting devices that permit the operator to preset the required depth of penetration of the cutting tool; and (5) coolant systems to provide lubrication and coolant to the cutting tool.

On other machines you can place the control levers in different positions; however, they serve the same purposes as those shown. Use the locking clamps to lock or "dog down" the table or head of a drill after it is positioned over the work. But make sure the locking action does not cause the drill or work to move slightly out of position.

TWIST DRILL

The twist drill is the tool generally used to drill holes in metal. This drill is formed either by forging and twisting grooves in a flat strip of steel or by milling a cylindrical piece of steel.

In figure 4-28 you see the principal parts of a twist drill: the body, the shank, and the point. The portion of the land behind the margin is relieved to provide body clearance. The body clearance helps reduce friction during drilling. The lip is the cutting edge, and the area called the lip clearance is on the cone of the drill. The dead center is the sharp edge located at the tip end of the drill. It is formed by the intersection of the cone-shaped surfaces of the point and should always be in the exact center of the axis of the drill. Do not confuse the point of the drill with the dead center. The point is the entire cone-shaped surface at the cutting end of the drill. The web of the drill is the metal column that separates the flutes. It runs the entire length of the body between the flutes and gradually increases in thickness toward the shank, giving additional rigidity to the drill.

The tang is found only on tapered-shank tools. It fits into a slot in the socket or spindle of the drill press and bears a portion of the driving strain. Its principal purpose is to make it easy to remove the drill from the socket with the aid of a drill drift. (NEVER use a file or screwdriver to do this job.)

The shank is the part of the drill that fits into the socket, spindle, or chuck of the drill press. The types of shanks that are most often found in Navy machine

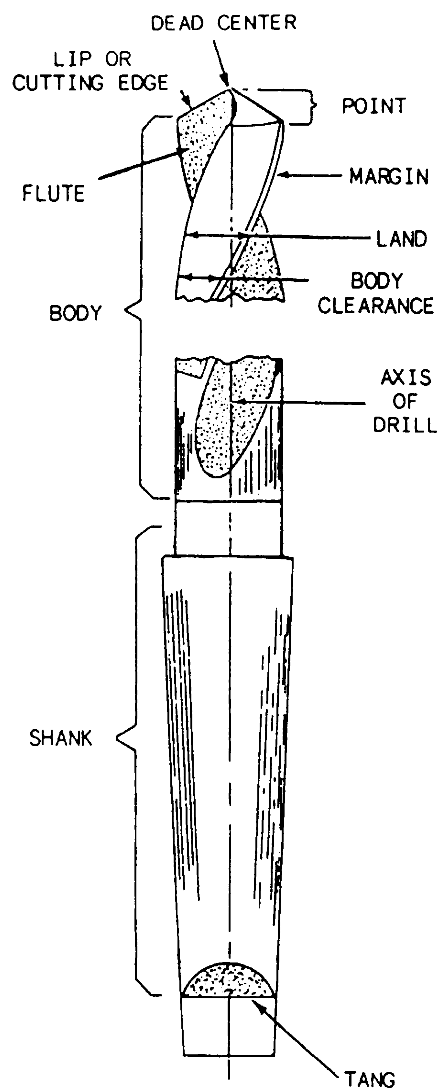


Figure 4-28.—The parts of a twist drill.

shops are the Morse taper shank, shown in figures 4-28 and 4-29A and the straight shank, shown in figures 4-29B and 4-29C.

Twist drills are made from several different materials. Drills made from high-carbon steel are available; however, their use is limited by the low cutting speed required to keep this type of drill from becoming permanently dull. Most of the twist drills that you will use are made from high-speed steel and will have two flutes (fig. 4-28).

Core drills (fig. 4-29A) have three or more flutes and are used to enlarge a cast or previously drilled hole. Core drills are more efficient and more accurate than the standard two-fluted drill. Core drills are made from high-speed steel.

A carbide-tipped drill (fig. 4-29B), is similar to a standard two-fluted drill with carbide inserts mounted along the lip or cutting edge. It is used to drill nonferrous metals, cast iron, and cast steel at high speeds. These drills are not designed for drilling steel and alloy metals.

A carbide-tipped die drill is also known as a spade drill (fig. 4-29C). It has two flutes that run parallel to the axis of the drill as opposed to the helical flutes of the standard two-fluted drill. It can be used to drill holes in hardened steel.

A standard two-fluted drill made from cobalt high-speed steel is superior in cutting efficiency and wear resistance to the high-speed steel drill. It is used at a cutting speed between the speed recommended for a high-speed steel drill and a carbide-tipped drill.

A solid carbide drill with two helical flutes is also available and can be used to drill holes in hard and abrasive metal where no sudden impact will be applied to the drill.

Drill sizes are indicated in three ways: by measurement, letter, and number. The nominal measurements range from 1/16 to 4 inches or larger, in 1/64-inch steps. The letter sizes run from “A” to “Z” (0.234 to 0.413 inch). The number sizes run from No. 80 to No. 1 (0.0135 to 0.228 inch).

Before putting a drill away, wipe it clean and then give it a light coating of oil. Do not leave drills in a place where they may be dropped or where heavy objects may fall on them. Do not place drills where they will rub against each other.

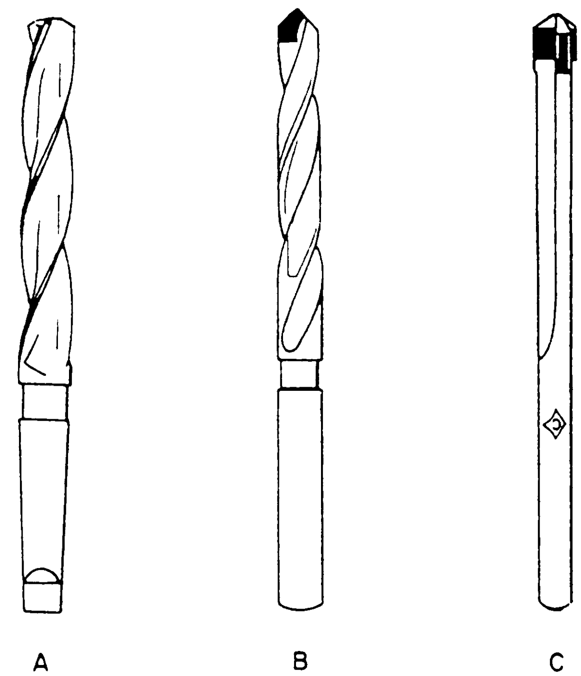


Figure 4-29.—Twist drills: A. Three-fluted core drill; B. Carbide tipped drill with two helical flutes; C. Carbide tipped die drill with two flutes parallel to the drill axis.

DRILLING OPERATIONS

Using the drill press is one of the first skills you will learn as a Machinery Repairman. Although a drill press is simpler to operate and understand than other machine tools in the shop, the requirements for accuracy and efficiency are no less strict. To achieve skill, you must know how to set feeds and speeds, how to hold the work, and how to ensure accuracy.

Speeds, Feeds, and Coolants

The cutting speed of a drill is expressed in feet per minute (fpm). This speed is computed by multiplying the circumference of the drill (in inches) by the revolutions per minute (rpm) of the drill. The result is then divided by 12. For example, a 1/2-inch drill turned at 100 rpm has a surface speed of 150 inches per minute. To obtain fpm, divide this figure by 12 which results in a cutting speed of approximately 12 1/2 feet per minute.

The correct cutting speed for a job depends on many factors. The main ones are: The machinability of a metal, any heat treatment process such as hardening, tempering, or normalizing, the type of drill used, the type and size of the drilling machine, the rigidity of the setup, the finish and accuracy required, and whether or not a cutting fluid is used. The following cutting speeds are recommended for high-speed steel twist drills. Carbon steel drills should be run at one-half these speeds, while carbide may be run at two to three times these speeds. As you gain experience, you will be able to vary the speeds to suit the job you are doing.

Low carbon steel	80-110 fpm
Medium carbon steel	70-80 fpm
Alloy steel	50-70 fpm
Corrosion-resistant steel (stainless)	30-40 fpm
Brass	200-300 fpm
Bronze	200-300 fpm
Monel	40-50 fpm
Aluminum	200-300 fpm
Cast iron	70-150 fpm

The speed of the drill press is given in rpm. Your shop will usually have tables giving the proper rpm at which to run a drill press for a particular metal or you may find them in machinists' handbooks. You can use a formula to determine the rpm required to give a specific rate of speed in fpm for a specific size drill. For example, if you wish to drill a hole 1 inch in diameter at the speed of 50 fpm, you would compute the rpm as follows:

$$rpm = \frac{fpm \times 12}{\pi \times D}$$

$$= \frac{50 \times 12}{3.1415 \times 1}$$

$$= \frac{600}{3.1416}$$

$$= 190$$

where

fpm = required speed in feet per minute

π = 3.1416

12 = constant

D = diameter of drillin inches

The feed of a drill is the rate of penetration into the work for each revolution. Feed is expressed in thousandths of an inch per revolution. In general, the larger the drill, the heavier the feed you may use. Always decrease feed pressure as the drill breaks through the bottom of the work to prevent drill breakage and rough edges. The rate of feed depends on the size of the drill, the material you are drilling, and the rigidity of the setup.

Use the following feed rates, given in thousandths of an inch per revolution (ipr), as a general guide until your experience allows you to determine the most efficient feed rate for each different job.

Drill Diameter	IPR
No. 80 to 1/8 inch	0.001-0.002
1/8 inch to 1/4 inch	0.002-0.004
1/4 inch to 1/2 inch	0.004-0.007
1/2 inch to 1 inch	0.007-0.015
Greater than 1 inch	0.015-0.025

Use the lower feed rate given for each range of drill sizes for the harder materials such as tool steel, corrosion-resistant steel and alloy steel. Use the higher feed rate for brass, bronze, aluminum, and other soft metals.

It is usually necessary to use a cutting oil or coolant to drill carbon steel, alloy steel, corrosion-resistant steel and certain nonferrous metals such as Monel. For most drilling operations, you can use soluble oil or a synthetic coolant. You may drill aluminum, brass, cast iron, bronze and similarly soft metals dry unless you use a high drilling speed and feed.

Holding the Work

Before drilling, be sure your work is well clamped down. On a sensitive drill press you will probably have to use a drill vise and center the work by hand. Because the work done on this drill press is comparatively light, the weight of the vise is sufficient to hold the work in place.

The larger drill presses have slotted tables upon which you may bolt or clamp work of considerable weight. Use T-bolts, which fit into the T-slots on the table, to secure the work. You can also use various

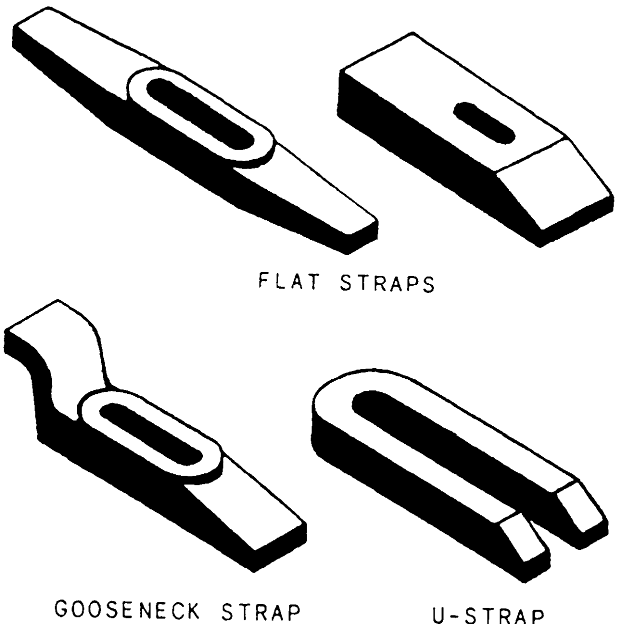


Figure 4-30.—Common types of clamping straps.

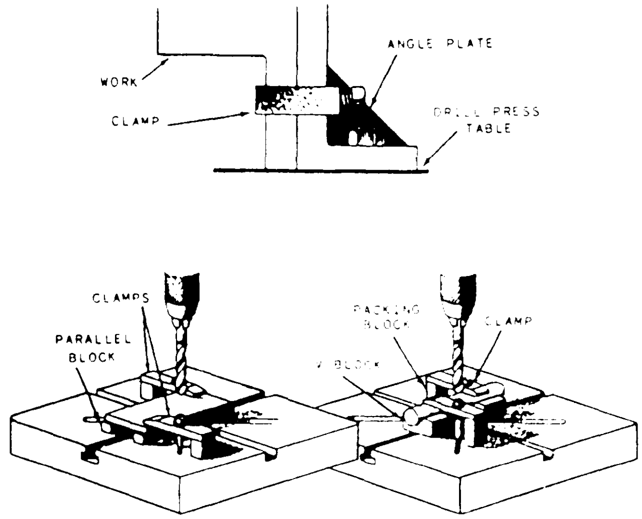


Figure 4-31.—Work mounted on the table.

types of clamping straps, shown in figure 4-30. (Clamping straps are also identified as clamps or dogs.) The U-strap is the most convenient for many setups because it has a larger range of adjustment.

It is often necessary to use tools such as steel parallels, V-blocks, and angle plates to support and hold the work. Use steel parallels to elevate the work above the table so you can better see the progress of the drill. Use V-blocks to support round stock, and angle plates to support work where you will drill a hole at an angle to another surface. Figure 4-31 shows some examples of setups.

Drilling Hints

To ensure accuracy in drilling, position the work accurately under the drill. Use the proper techniques to prevent the drill from starting off center or from moving out of alignment during the cut. Here are some hints that will help you to correctly start and complete a drilling job.

- Before setting up the machine, wipe all foreign matter from the spindle and the table. A chip in the spindle socket will cause the drill to have a wobbling effect which tends to make the hole larger than the drill. Foreign matter on the work holding device under the workpiece tilts it in relation to the spindle, causing the hole to be out of alignment.

- Center punch the work at the point to be drilled. Position the center-punched workpiece under the drill. Use a dead center inserted in the spindle socket to align the center-punch mark on the workpiece directly under the axis of the spindle.

- Bring the spindle with the inserted center down to the center-punch mark and hold it in place lightly while you fasten the locking clamps or dogs. This will prevent slight movement of the workpiece, table, or both when they are clamped in position.

- Insert a center drill (fig. 4-32) in the spindle and make a center hole to aid in starting the drill. This is not necessary on small drills on which the dead center of the drill is smaller than the center-punch mark. But on large drills it will prevent the drill from “walking’* away from the center-punch mark. This operation is especially important to drill holes on curved surfaces.

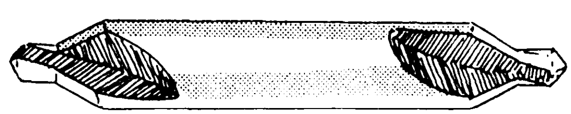


Figure 4-32.—Combined drill and countersink (center drill).

- If you use a drill smaller than the required size to make a pilot hole, it will increase accuracy by eliminating the need to do any cutting with the dead center of the finishing drill. This will decrease the pressure required to feed the finishing drill, and decrease the width of cut taken by each drill. In drilling holes over 1 inch in diameter, you may need to use more than one size of pilot drill to increase the size of the hole by steps until you reach the finished size.

- If the outer corners of the drill (margin) appear to be wearing too fast or have a burnt look, the drill is going too fast.

- If the cutting edges (lips) chip during drilling, you have ground too much lip clearance into the drill, or you are using too heavy a feed rate.

- A very small drill will break easily if the drill is not going fast enough.

- When you drill a hole that is more than three or four times the drill diameter in depth, back out the drill frequently to clear the chips from the flutes.

- If the drill becomes hot quickly, is difficult to feed, squeals when being fed and produces a rough finish in the hole, it has become dull and requires resharpening.

- If the drill has cutting edges of different angles or unequal length, the drill will cut with only one lip and will wobble in operation. This will produce an oversized hole.

- If the drill will not penetrate the work, you have ground insufficient or no lip clearance into the drill.

- The majority of drilled holes will be oversized regardless of the care you take to ensure a good setup. Generally, you can expect the oversize to average an amount equal to 0.004 inch times the drill diameter plus 0.003 inch. For example, you can expect a 1/2-inch drill to produce a hole approximately 0.505 in diameter ([0.004 X 0.500] + 0.003). This amount can vary up or down depending on the condition of the drilling machine and the twist drill.

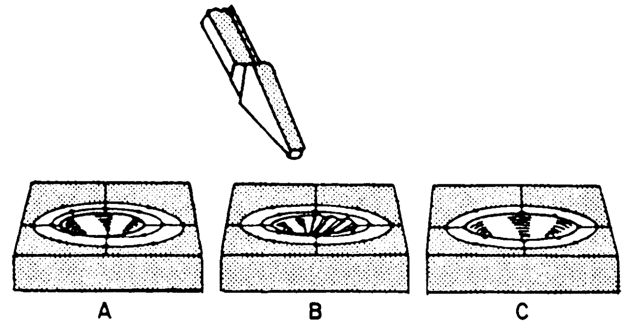


Figure 4-33.—Using a half-round chisel to guide a drill to the correct center.

Correcting Offcenter Starts

A drill may start off center because of improper center drilling, careless starting of the drill, improper grinding of the drill point, or hard spots in the metal. To correct this condition, (See fig. 4-33.) take a half-round chisel and cut a groove on the side of the hole toward which the center is to be drawn. The depth of this groove depends upon the eccentricity (deviation from center) of the partially drilled hole with the hole to be drilled. When you have drilled out the groove, lift the drill from the work and check the hole for concentricity with the layout line. Repeat the operation until the edge of the hole and the layout line are concentric.

When you use this method to correct an off center condition, be very careful that the cutting edge or lip of the drill does not grab in the chisel groove. Generally, you should use very light feeds until you establish the new center point. (Heavy feeds cause a sudden bite in the groove which may push the work out of the holding device, or break the drill.)

Counterboring, Countersinking, and Spotfacing

A counterbore is a drilling tool used in the drill press to enlarge portions of previously drilled holes. The purpose is to allow the heads of fastening devices to be flush with or below the surface of the workpiece. The parts of a counterbore that distinguish it from a regular drill are a pilot, which aligns the tool in the hole to be counterbored, and the cutting edge of the counterbore, which is flat so that a flat surface is left at the bottom of the cut, enabling fastening devices to seat flat against the bottom of the counterbored hole.

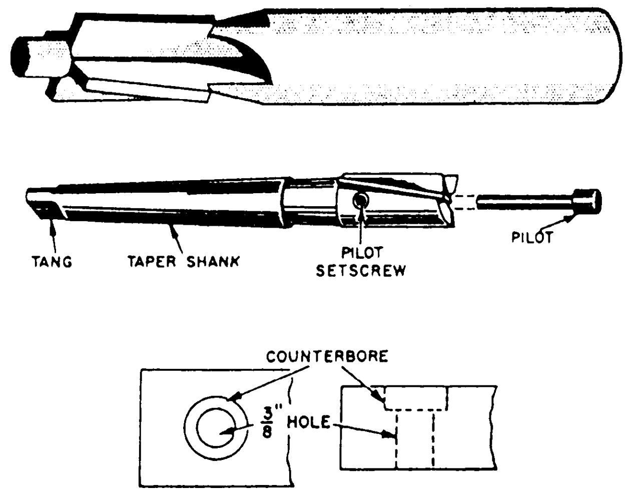


Figure 4-34.—Two types of counterbore.

Figure 4-34 shows two types of counterbores and an example of a counterbored hole. The basic difference between the counterbores illustrated is that one has a removable pilot and the other does not. You can use a conterbore with a removable pilot to counterbore a range of hole sizes by simply using the appropriate size pilot. A counterbore with a fixed pilot may be used only with holes of the same dimensions as the pilot.

Countersinks are used to seat flathead screws flush with the surface. The basic difference between countersinking and counterboring is that a countersink makes an angular sided recess, while the counterbore forms straight sides. The angular point of the countersink acts as a guide to center the tool in the hole being countersunk. Figure 4-35 shows two common types of countersinks.

Spotfacing is an operation that cleans up the surface around a hole so that a fastening device can be seated flat on the surface. This operation is commonly required on rough surfaces that have not been machined and on the circumference of concave or convex workpieces. Figure 4-36 shows an

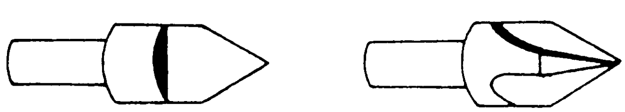


Figure 4-35.—Countersinks.

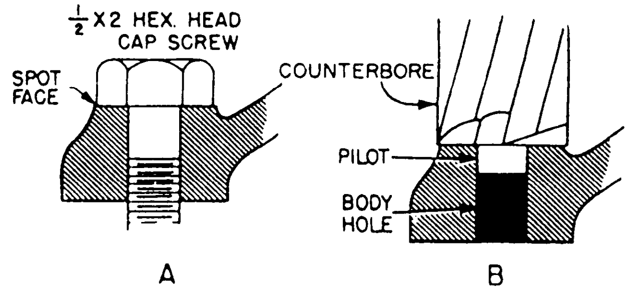


Figure 4-36.—Examples of spotfacing.

example of spotfacing and its use with fastening devices. You will usually do this with a counterbore.

Reaming

The drill press also may be used for reaming. For example, when specifications call for close tolerances, the hole must be drilled slightly undersize and then reamed to the exact dimension. Reaming also removes burrs in a drilled hole or enlarges a previously used hole for new applications.

Machine reamers have tapered shanks that fit the drilling machine spindle. Be sure not to confuse them with hand reamers, which have straight shanks. You will ruin hand reamers if you use them in a machine.

There are many types of reamers, but the ones used most extensively are the straight-fluted, the taper pin, and the expansion types. They are illustrated in figure 4-37.

The straight-fluted reamer is made to remove small portions of metal and to cut along the edges to bring a hole to close tolerance. Each tooth has a rake angle that is comparable to that on a lathe tool.

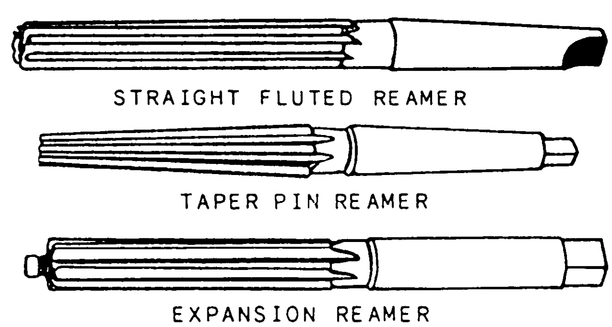


Figure 4-37.—Reamers.

The taper pin reamer has a tapered body and is used to smooth and true tapered holes and recesses. The taper pin reamer is tapered at 1/4 inch per foot.

The expansion reamer is especially useful to enlarge reamed holes by a few thousandths of an inch. It has a threaded plug in the lower end which expands the reamer to various sizes.

To ream a hole, follow the steps outlined below:

1. Drill the hole about 1/64 inch less than the reamer size.
2. Substitute the reamer in the drill press without removing the work or changing the position of the work.
3. Adjust the machine for the proper spindle speed. (Reamers should turn at about one-half the speed of the twist drill.)
4. Use a cutting oil to ream. Use just enough pressure to keep the reamer feeding into the work; excessive feed may cause the reamer to dig in and break.
5. The starting end of a reamer is slightly tapered; always run it all the way through the hole. **NEVER RUN A REAMER BACKWARD** because the edges are likely to break.

Tapping

Special attachments allow you to cut internal screw threads with a tap driven by the drilling machine spindle. They can save considerable time when you must thread a number of identically sized holes. The attachment is equipped with a reversing device that automatically changes the direction of rotation of the tap when either the tap strikes the bottom of the hole or a slight upward pressure is applied to the spindle down-feed lever. The reversing action takes place rapidly, permitting accurate control over the depth of the threads being cut. A spiral-fluted tap should be used to tap a through hole while a standard straight-fluted plug tap can be used in a blind hole. A good cutting oil should always be used in tapping with a machine.

DRILLING ANGULAR HOLES

An angular hole is a hole with a series of straight sides of equal length. A square (4-sided), a hexagon (6-sided), a pentagon (5-sided), and an octagon 8-sided) are examples of angular holes. You can use a broach to make an angular hole that goes all the way through a part. However, you will need an angular drill to make a blind hole, one in which the angular hole does not go all the way through the part.

ANGULAR DRILL EQUIPMENT

The equipment required to drill angular holes is specialized and is designed to do only this particular operation. The following paragraphs contain a brief description of the equipment. A complete description of the equipment and its use is available from the manufacturer when the equipment is ordered.

Chuck

The chuck (fig. 4-38A) used to drill angular holes is of an unusual design. While it holds the drill in a position parallel to the spindle of the lathe or drill press and prevents it from revolving, it allows the drill to float freely so that the flutes can follow the sides of the angular hole in the guide plate. The chuck is available with a Morse taper shank to fit most lathes and drill presses. There are several different sizes of chucks, each capable of accepting drills for a given range of hole sizes.

Guide Plates

The guide plate (fig. 4-38B) is the device that causes the drill to make an angular hole. The free-floating action of the chuck allows the drill to

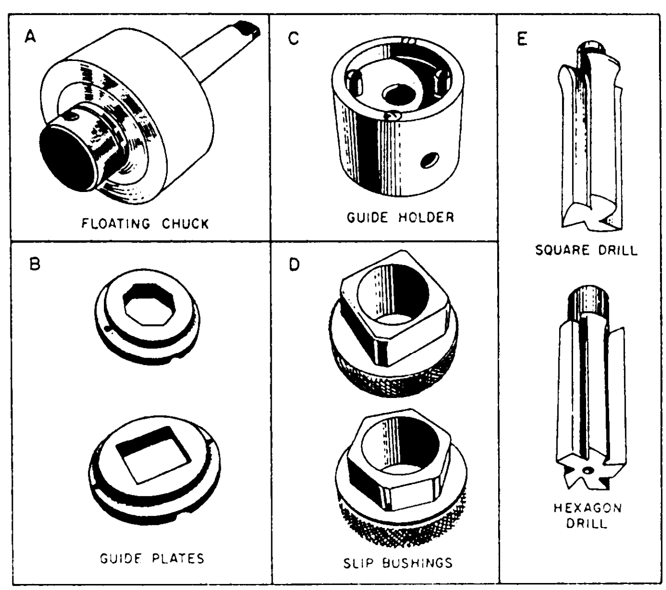


Figure 4-38.—Equipment for drilling angular holes. A. Chuck; B. Guide plate; C. Guide holder; D. Slip bushing; E. Angular drill.

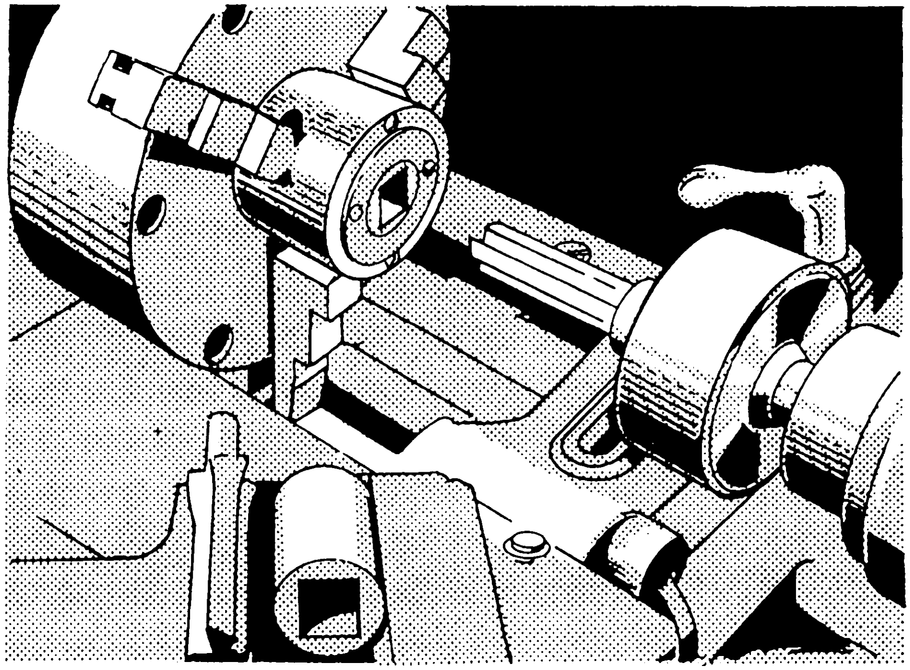


Figure 4-39.—Lathe setup for drilling an angular hole.

randomly follow the straight sides and corners of the guide plate as it is fed into the work. Attach the guide plate to a guide holder when you use a lathe and directly to the work when you use a drill press. A separate guide plate is required for each different shape and size hole.

Guide Holder

The guide holder (fig. 4-38C), as previously stated, holds the guide plate and is placed over the outside diameter of the work and locked in place with a setscrew. Use the guide holder when you are doing the work in a lathe; it is not required for drill press operations.

Slip Bushings

Before you drill with the angular hole drill, you must drill a normal round hole in the center of the location where the angular hole will be located. This pilot hole reduces the pressure that would otherwise be required to feed the angular drill and ensures that the angular drill will accurately follow the guide plate. In a lathe, you need only drill a hole using the tailstock since it and the chuck will automatically center the pilot hole. In a drill press, you must devise a method to help you align the pilot hole. A slip bushing will do the job quickly and accurately. The slip bushing (fig. 4-38D) fits into the guide plate. It has a center hole which is the correct size for the pilot hole of the particular size angular hole being drilled. After you have installed the bushing, position the correct drill so that it enters the hole in the slip bushing and drill the pilot hole.

Angular Drill

The angular drills (fig. 4-38E) are straight fluted and have one less flute or cutting lip than the number of sides in the angular hole they are designed to drill. The drills have straight shanks with flats machined on them to permit securing them in the floating chuck with setscrews. The cutting action of the drill is made by the cutting lips or edges on the front of the drill.

OPERATION

The procedure to drill an angular hole is similar to that to drill a normal hole, differing only in the preliminary steps required in setting the job up. The feeds and speeds used to drill angular holes should be slower than those recommended to drill a round hole of the same size. Obtain specific recommendations concerning feeds and speeds from the information provided by the manufacturer. Use a coolant to keep the drill cool and help flush away the chips.

The following procedures apply when you do the work on a lathe. See figure 4-39 for an example of a lathe setup.

1. Place the work to be drilled in the lathe chuck. The work must have a cylindrical outside diameter and the intended location of the angular hole must be in the center of the work.

2. Place the guide holder over the outside diameter of the work and tighten the setscrew. If the bore in the back of the guide holder is larger than the diameter of the work, make a sleeve to adapt the two together. If the part to be drilled is short, place it in the guide holder and place the guide holder in the chuck.

3. Drill the pilot hole at this time. The size of the pilot hole should be slightly smaller than the distance across the flats of the angular hole. The manufacturer makes specific recommendations on pilot hole sizes.

4. Attach the guide plate to the guide holder.

5. Mount the floating chuck in the lathe tailstock spindle and place the drill in the chuck. Tighten the setscrews to hold the drill securely.

6. You are now ready to drill the angular hole. Do not force the drill into the work too rapidly, and use plenty of coolant.

The following procedures apply when you drill an angular hole on a drill press. Clamp the guide plate directly to the work and drill the pilot hole by using a slip bushing placed in the guide plate to ensure alignment. Once you have positioned the work under the drill press spindle and have drilled the pilot hole, do not move the setup. Any movement will cause misalignment between the work and the angular drill.